

1991

NASA/ASEE SUMMER FACULTY FELLOWSHIP PROGRAM

MARSHALL SPACE FLIGHT CENTER
THE UNIVERSITY OF ALABAMA

Development of a Pressure Based Multigrid
Solution Method for Complex Fluid Flows

Prepared By:
Academic Rank:
Institution:

Wei Shyy
Associate Professor
University of Florida
Department of Aerospace Engineering,
Mechanics and Engineering Science

NASA/MSFC:

Office:
Division:
Branch:

Structures and Dynamics Laboratory
Aerophysics
Computational Fluid Dynamics

NSFC Colleague:

Mr. P. Kevin Tucker

Contract No.:

NGT-01-008-021
The University of Alabama

In spite of the rapid advent of computer technology over the past several decades, in order to be able to deal accurately with the complex flow problems, there is a need for continuous improvement of computational efficiency. With the large number of grid points normally required for practically relevant flows, iterative methods are often used. However, it is well known that for a SOR type of method performed on a single-grid (SG) system, the number of iterations required typically increases in proportion to the number of grid points. Since the CPU time on a per iteration basis scales with the number of grid points, the grid resolution typically required for satisfactorily solving an incompressible recirculating flow problem would need substantial computational time even on supercomputers. In order to reduce this difficulty associated with the SG solution procedure, the multigrid (MG) technique has been identified as a very useful means for improving the convergence rate of iterative methods.

The multigrid technique, originally developed for the efficient solution of linear, elliptic differential equations, has been used in the field of computational fluid dynamics with increasing popularity. Successes have been reported for both compressible and incompressible flows. However, it is not until more recently that attempts have been reported, with varying degrees of success, on the application to incompressible recirculating flows with primitive variables. Substantial differences exist between the basic numerical algorithms embodied in those studies, notably the way the pressure field is obtained, the choice of the convection scheme, and the design of the grid layout for the velocity and pressure variables. Generally speaking, two methods have been developed for obtaining the solution field, namely, the decoupled method, and the coupled method. In the decoupled method, a two-level iterative procedure is usually employed; one is the outer iteration needed to progressively update different partial differential equations, and the other is the inner iteration devised to solve the system of linear algebraic equations resulting from the discretization procedures of each partial differential equation with other variables remaining unchanged. Within the outer iteration, for a two-dimensional flow with u , v , and p as the dependent variables for example, a cyclic outer iterative procedure is designed to sequentially solve, say, the linearized x -momentum equation first, the linearized y -momentum equation next, and the pressure (correction) equation last. After sweeping through all three equations to obtain partially converged solutions, the x -momentum equation is again invoked to initiate a new cycle, until all three equations are satisfactorily solved. Within the inner iteration, say, the x -momentum equation is discretized and linearized, and the resulting set of linear equations is then solved by an iterative procedure such as the LSOR method till the prescribed number of iterations or the convergence criterion has been reached. For a decoupled algorithm, the treatment of coupling among the dependent variables, such as velocity and pressure, is critical to the overall convergence. Hence, the performance of this type of method depends on such factors as the Reynolds number and the distribution and skewness of the grid in sensitive manners.

In contrast, a coupled method, which solves the velocity vector and the scalar variables at a point, line, or plane simultaneously, usually shows robust performance with respect to parameters such as the Reynolds number. In the context of Cartesian coordinates, the

coupled method is also found to be relatively insensitive to the number of the grid points employed. With the use of curvilinear coordinates, however, the situation is not as favorable, since, depending on the characteristics of the grid skewness, one either has to treat the cross-derivative terms explicitly as source term or solve equations whose coefficient matrix is no longer sparse. Furthermore, from the viewpoint of developing generic computational capabilities for flows involving different physical mechanisms such as turbulence, heat transfer, combustion, and phase change, it is preferable that one does not have to redo the algorithm for a different number of partial differential equations. In this regard, the decoupled method has a clear advantage since it can handle a different number of equations in more a flexible manner. It is with this motivation that in the present work a multigrid method in curvilinear coordinates is developed in conjunction with a basic flow solver reported in Refs [1,2], which utilizes a decoupled algorithm to solve incompressible recirculating flow problems.

A full multigrid /full approximation storage (FMG/FAS) algorithm is utilized to solve the incompressible recirculating flow problems in complex geometries. The algorithm is implemented in conjunction with a pressure-correction/staggered-grid type of technique using the curvilinear coordinates. In order to illustrate the performance of the method, two flow configurations, one a square cavity driven by a sliding top wall and the other a channel with multiple bumps are used as the test problems. Comparisons are made between the performances of the multigrid and single-grid methods, measured by the number of fine grid iterations, equivalent work units, and CPU time. Besides demonstrating that the multigrid method can yield substantial speed-up with wide variations in Reynolds number, grid distributions, and geometry, issues such as the convergence characteristics at different grid levels, the choice of convection schemes, and the effectiveness of the basic iterative smoothers are studied. An adaptive grid scheme is also combined with the multigrid procedure to explore the effects of grid resolution on the multigrid convergence rate as well as the numerical accuracy. A full account of the technique developed along with illustrative results can be found in Ref [3]. Two figures shown here give a depiction of the grid cycle adopted and a sample of the comparison of the relative performance of the method developed.

References

- [1] W. Shyy, S.S. Tong and S.M. Correa, "Numerical Recirculating Flow Calculation Using Body-Fitted Coordinate System," Numer. Heat Transf., vol. 8, 1985, pp. 99-113.
- [2] M.E. Braaten and W. Shyy, "A Study of Recirculating Flow Computation Using Body-Fitted Coordinates: Consistency Aspects and Mesh Skewness," Numer. Heat Transf., Vol. 9, 1986, pp. 559-574.
- [3] W. Shyy and C.-S. Sun, "Development of a Pressure-Correction/Staggered-Grid Based Multigrid Solver for Computing Incompressible Recirculating Flows," AeMES Technical Report, University of Florida, Dept. of Aerospace Engineering, Mechanics and Engineering Science, 1991.

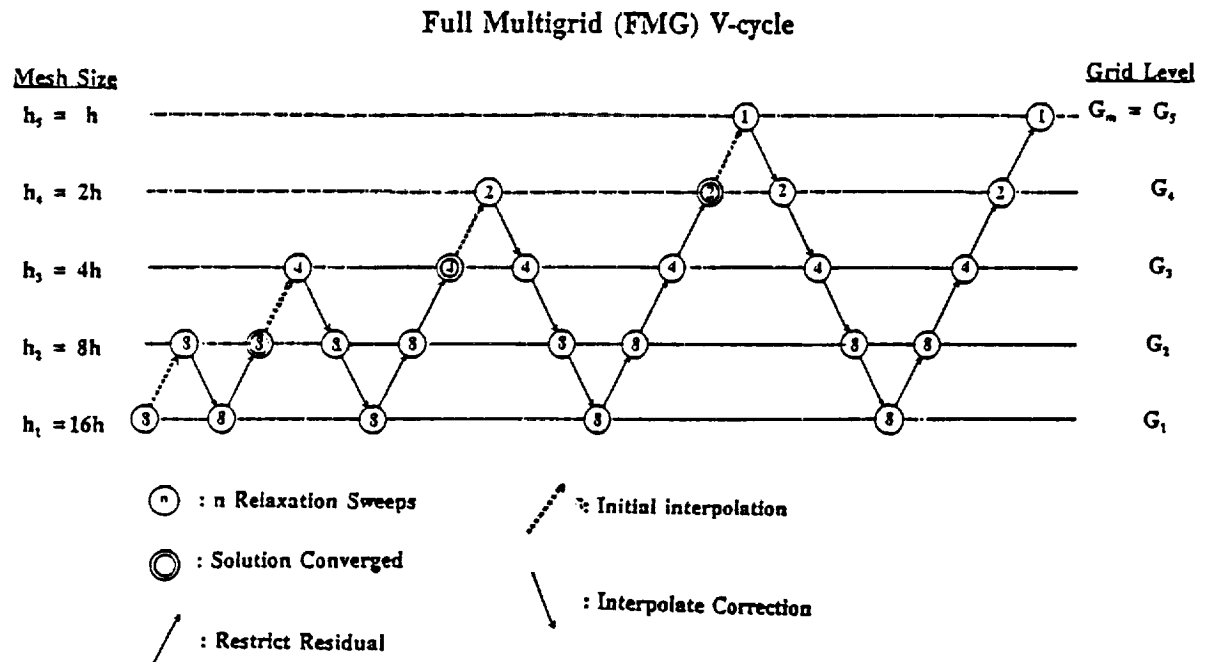
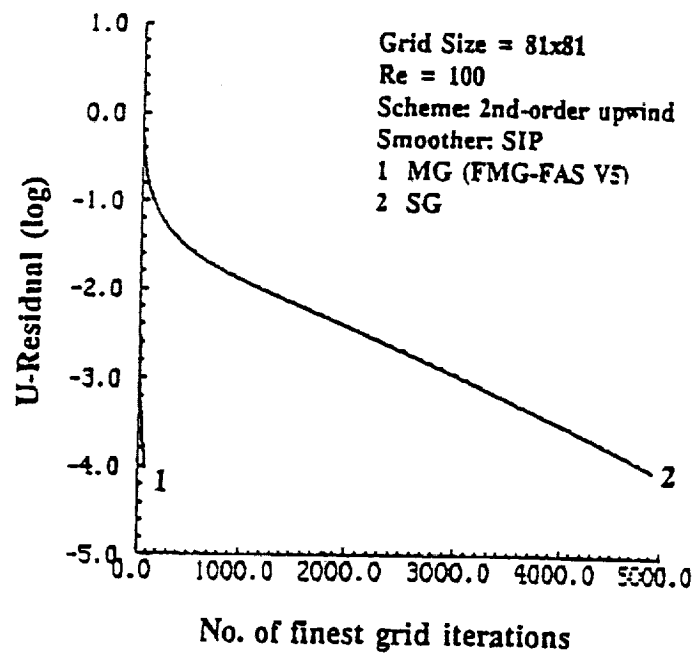
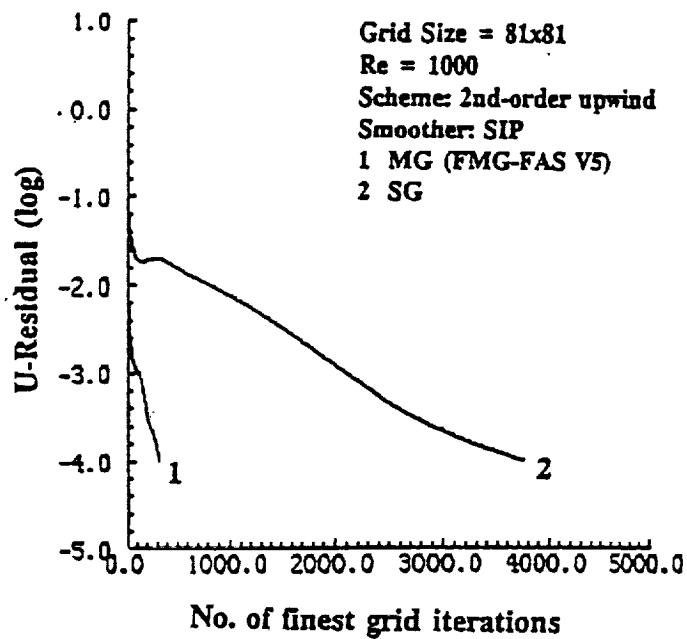


Fig. 1 A full multigrid (FMG) procedure with fixed V-cycles
M = 5 grid levels.



(a) Re = 100



(b) Re = 1000

Fig. 2 Comparison of convergence rates between multigrid and single-grid solvers on 81x81 grid with SIP as smoother for cavity flow with (a) Re = 100, (b) Re = 1000, and second-order upwind convection scheme.